

Sheet List

main

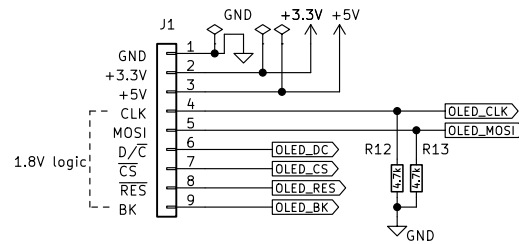


File: main.kicad_sch

Update History

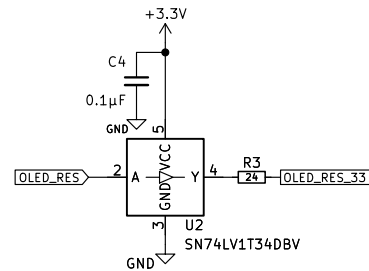
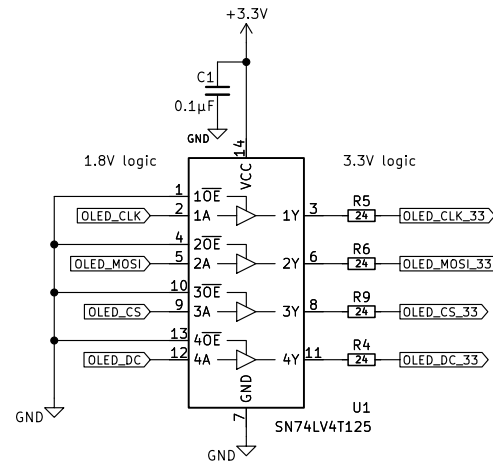
Rev.No	Date	Comments
A1	2021-04-03	First revision.
B1	2022-01-23	Changed LCD to 1.9inch
B2	2022-11-11	Changed FPC Conn
B3	2022-11-25	No schematic changed. Changed FPC Conn position.

IN

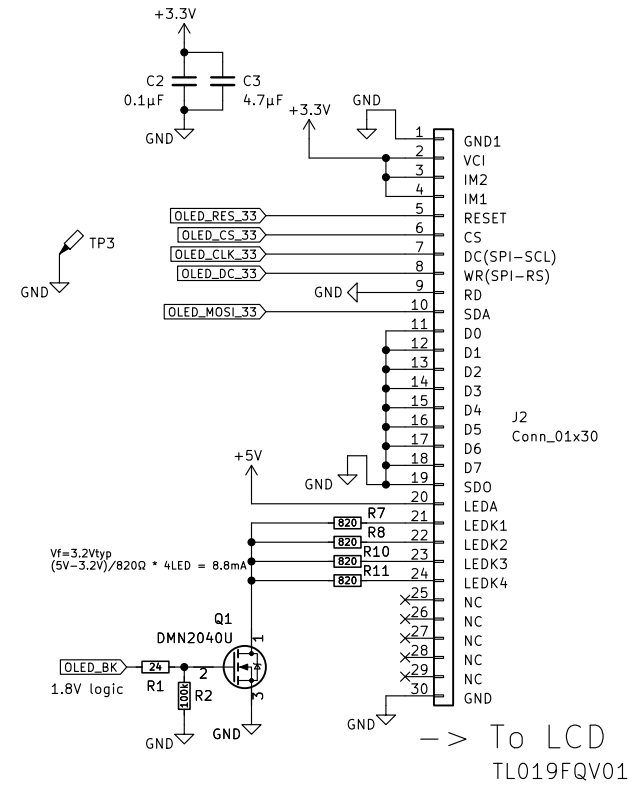


From Main PCB ->

Buffer



OUT



Misc.

Mounting hole
M2x2